

# SolidWorks Workshop Institute of Field roBOTics

## Revolved Feature

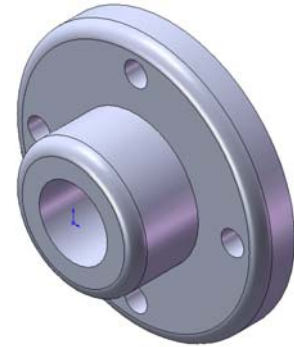
### Exercise 1: Hub/Flange

Create this part using the dimensions provided.  
Use relations and equations where applicable to maintain the design intent.

This lab uses the following skills:

- Revolved features
- Circular patterning

Units: **inches**

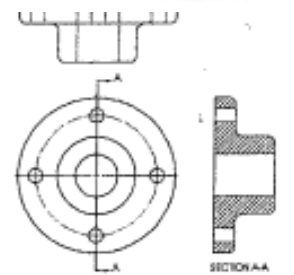


#### Design Intent

The design intent for this part is as follows:

1. Holes in the pattern are equally spaced.
2. Holes are equal diameter.
3. All fillets are equal and are **R0.25"**.

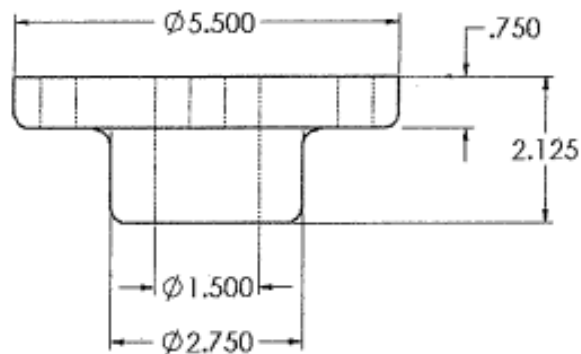
Note that construction circles can be created using the **Properties** of a circle.



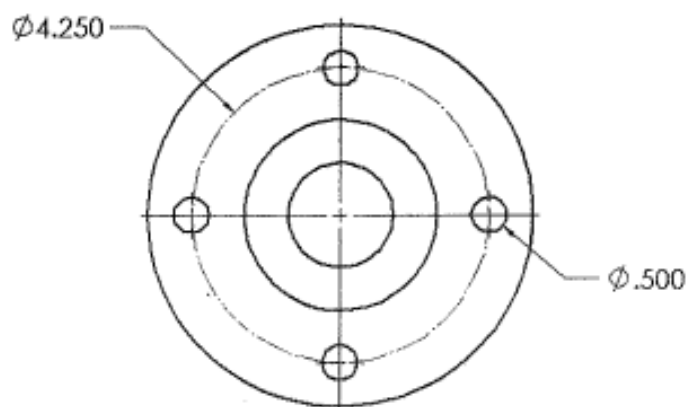
#### Dimensioned Views

Use the following graphics with the description of the design intent to create the part.

Top View



Front View



## Exercise 2: Hand Wheel

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### Case Study: Handwheel



#### Stages in the Process

Some key stages in the modeling process of this part are shown in the following list.

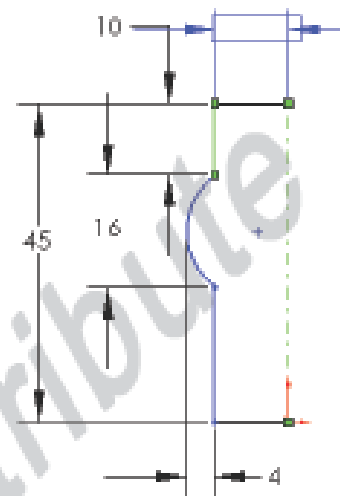
- **Design intent**  
The part's design intent is outlined and explained.
- **Revolved features**  
The center of the part is the Hub, a revolved shape. It will be created from a sketch with a construction line as the axis of revolution.
- **Multibody solids**  
Create two discrete solids, the Hub and the Rim, connecting and merging them using a third solid, the Spoke.
- **Sweep features**  
The Spoke feature is created using a sweep feature, a combination of two sketches that define a sweep profile moving along a sweep path.
- **Circular patterns**  
Rather than model the same spoke multiple times, we will create a pattern of evenly spaced Spokes around the centerline of the Hub.
- **Analysis**  
Using tools that are included in the SolidWorks software, you can perform basic analysis functions such as mass properties calculations and first-pass stress analysis. Based on the results, you can make changes to the part's design.

**9 Dimension to centerline.**

Dimension between the centerline and the outer vertical edge to create a horizontal linear dimension.

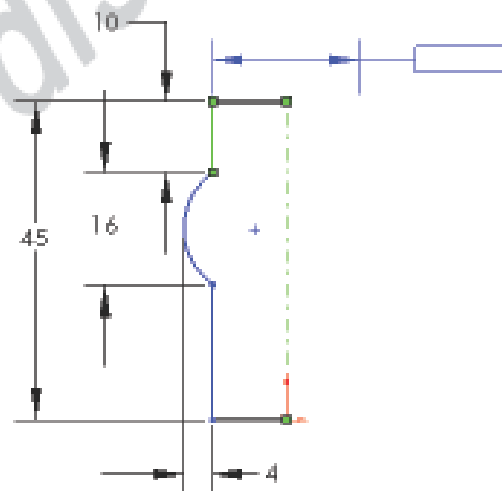
*Do not* click to place the dimension text just yet.

Notice the preview. If you place the text now, you will get a radius dimension.



**10 Move the cursor.**

Move the cursor to the right of the centerline. The preview changes to a diameter dimension.

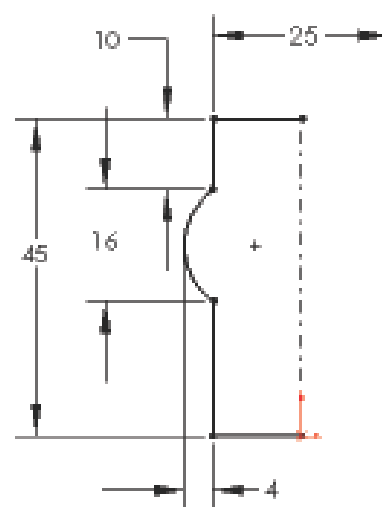


**11 Resulting dimension.**

Click to place the dimension text. Change the value to **25mm** and press **Enter**.

Normally, a diameter dimension should have a diameter symbol preceding it thus:

Ø 25. When the revolved feature is created from the sketch, the system will automatically add the diameter symbol to the **25mm** dimension.



**12 Make the feature.**

Click **Boss/Base, Revolve...** from the **Insert** menu. A message will appear indicating that the sketch is an open contour and asking if you want to close the contour automatically. Click **Yes**.

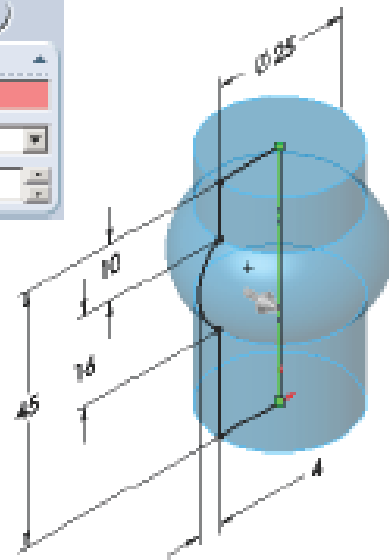
The PropertyManager appears with these default end conditions:



**One Direction**

**Angle 360°**

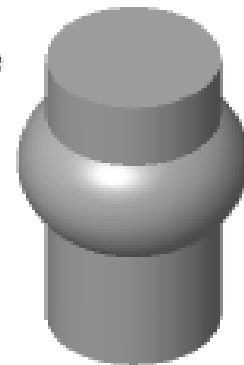
Accept these defaults by clicking **OK**.



**13 Finished feature.**

The solid revolved feature is created as the first feature of the part.

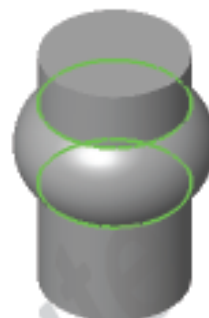
Rename it **Hub**.



#### 14 Edit the sketch.

Right-click the Hub and select **Edit Sketch**.


You can also right-click the feature in the FeatureManager design tree and achieve the same result.



#### 15 Normal To.

Click **Normal To**  on the Standard Views toolbar to change the view so you can see it true size and shape.

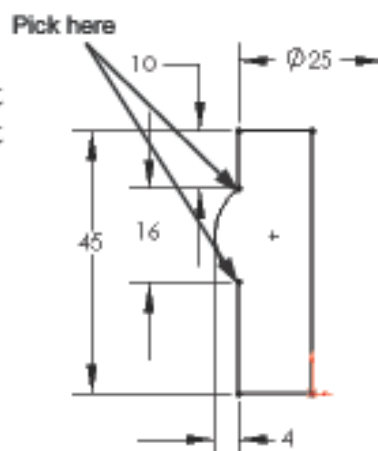
#### 16 Fillet settings.

Select the  tool and set the value to 5mm. Make sure the **Keep constrained corners** option is checked.



#### 17 Selections.

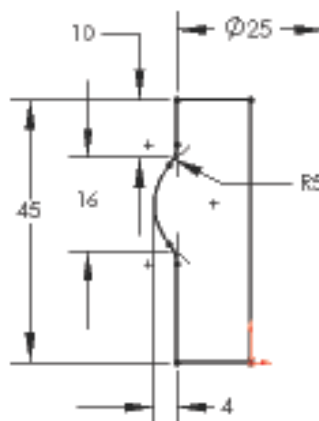
Select both endpoints of the arc, as indicated. When each is selected, the fillet will appear. The dimension drives both but only appears once, at the first selection.




Since the endpoints that were filleted had dimensions, **Virtual Sharp** symbols are added where the corners were. These symbols represent the missing corners and can be dimensioned to or used within relations.

Notice the 25mm dimension. As mentioned in step 11 on page 183, a diameter symbol now precedes the dimension.

Close the PropertyManager.



**18 Rebuild the model.**

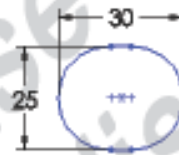
To cause the changes to take effect click the **Rebuild**  tool.



**Building the Rim**

The Rim of the Hand-wheel is another revolved feature. It too is revolved 360°. The profile of the Rim is an oval shape, made up of two 180° arcs and two lines.

The Rim will be created as a separate solid body, not merged to the Hub.



**19 Sketch.**

Create a new sketch on the **Right** reference plane. Orient the model in the same direction.

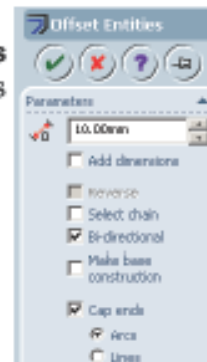
**20 Horizontal centerline.**

Sketch a short horizontal centerline somewhere off of the model.



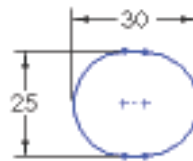
**21 Offset with cap ends.**

The **Offset Entities** tool has a **Cap ends** option that creates **Arcs** to close the ends of a **Bi-directional** offset. Select the centerline and offset using the options shown.




## 22 Add dimensions.

Dimension the sketch as shown in the illustration at the right.



The sketch entity **Point** can be used to locate a position in a sketch that other geometry (endpoints for example) cannot.


- Click **Point**  on the Sketch toolbar.
- Or from the **Tools** menu, click **Sketch Entities, Point**.

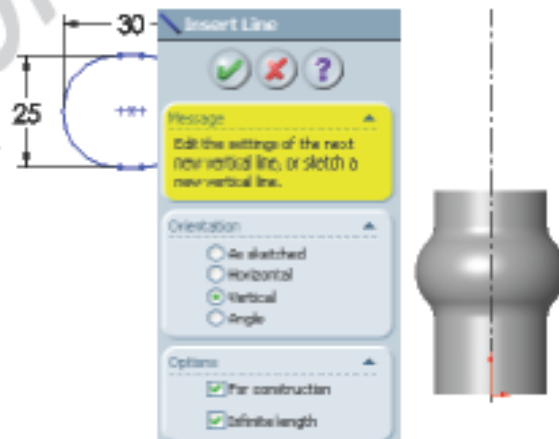
## 23 Add a point.

Click **Point**  and add a point at the midpoint of the centerline.



## 24 Rotation axis.

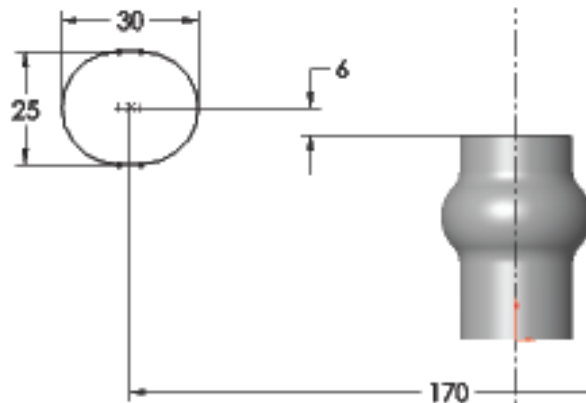
Add a centerline using the **Centerline**  tool, setting **Vertical** and **Infinite length**. Place the line at the origin. This will be the axis of revolution for the revolved feature.



## 25 Add dimensions.

Add dimensions from the centerline to the point and the arc center to the Hub edge.

The sketch is now fully defined.

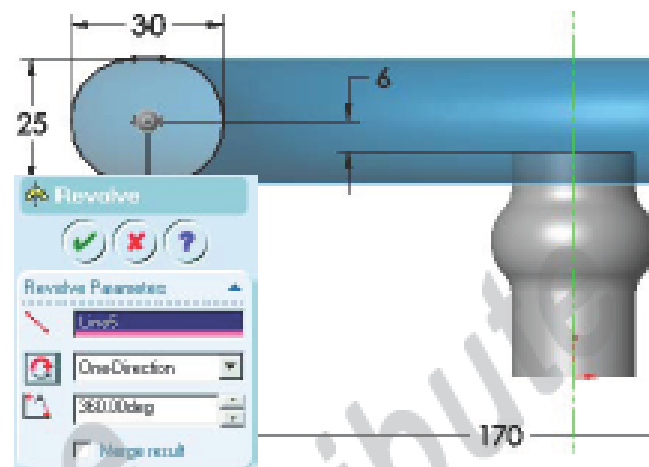


## 26 Potential ambiguity.

This sketch contains two centerlines. The system will not know which centerline is intended to be the axis of revolution. The centerline to be used can be selected either before or after selecting the **Revolve** tool

**27 Completed feature.**

Select the infinite vertical centerline. From the **Insert** menu, choose **Boss/Base, Revolve....** Use an angle of **360°**. Rename the feature to **Rim**.



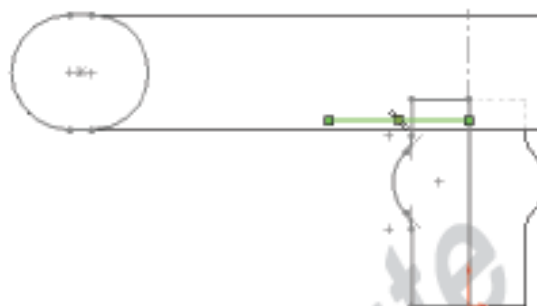
**28 Setup.**

Setup for sketching:

- Create a new sketch using the **Right** reference plane.
- Show the sketches of the **Hub** and **Rim**.
- Change the display to **Hidden Lines Visible**.

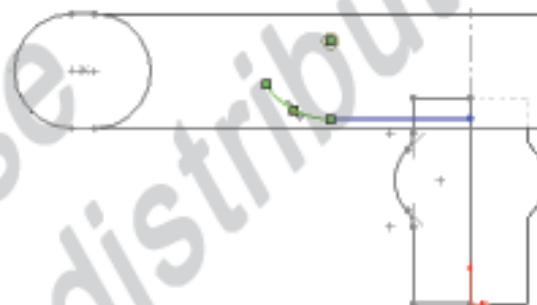
**29 Sketch line.**

Sketch a horizontal **Line** running from the centerline inside the Hub boundaries.



**30 Tangent arc.**

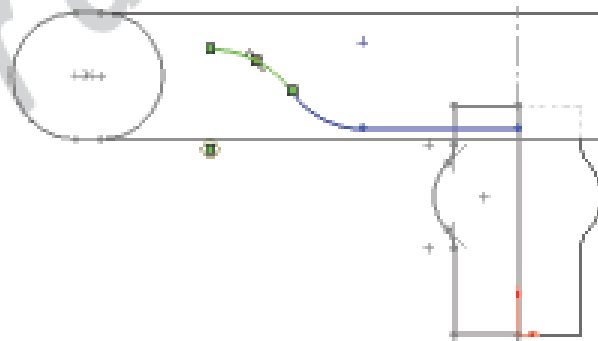
Create a **Tangent Arc** from the line endpoint in the direction shown. The actual values are not important as you sketch. They will be defined by dimensions later.





**31 Connecting tangent arc.**

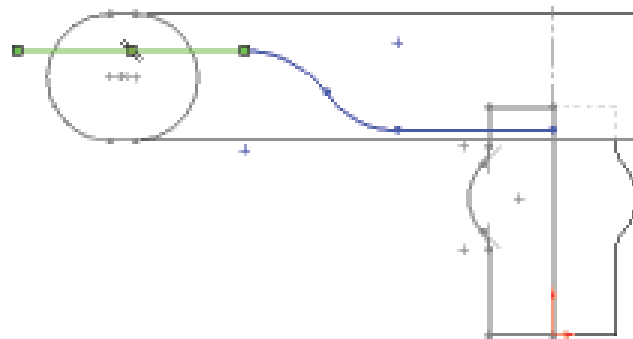
With **Tangent Arc** still selected, continue sketching by using the previous arc's endpoint as a start. Sketch this arc tangent to the first, ending at a horizontal tangency position.



When the vertical inference line coincides with the arc's center, the tangent of the arc is horizontal.

**32 Horizontal line.**

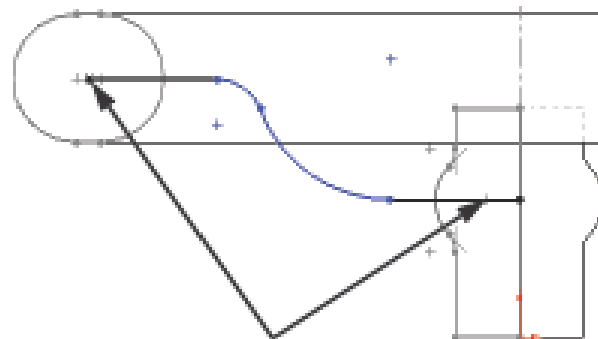
Sketch a final **Line**. It is horizontal, with its length to be determined by dimensioning.



**33 Relations.**

Drag and drop the left endpoint of the line onto the point of the **Rim** sketch. A **Coincident** relation is added.

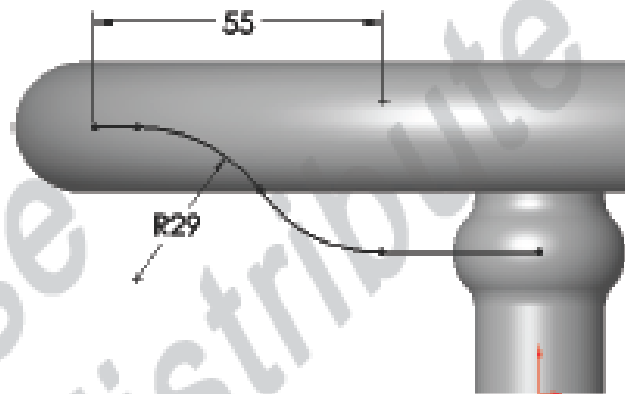
Add another relation between the line at the opposite end and the centerpoint of the arc.



The geometry sketched will act as the “centerline” for the profile sketch.

**35 Add dimensions.**

Add an **Equal** relation to the arcs. Dimensions are added to define the shape. Picking end points and center points allows for more options when the creating the dimensions.

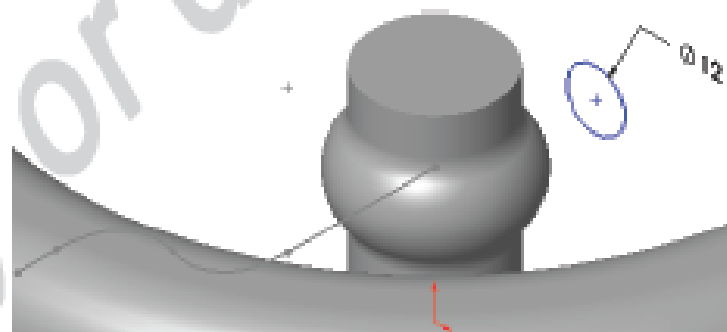


**36 Exit sketch.**

Right-click in the sketch and choose **Exit Sketch** to close the sketch without using it in a feature.

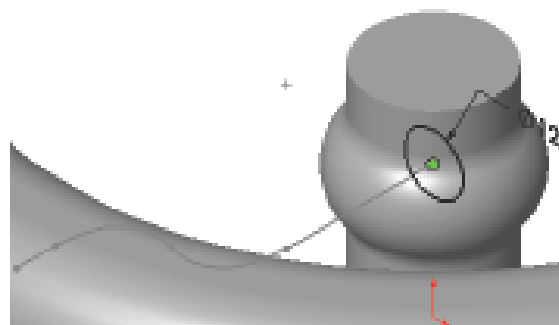
**37 Profile.**

Sketch a circle on the **Front** plane and dimension the diameter.



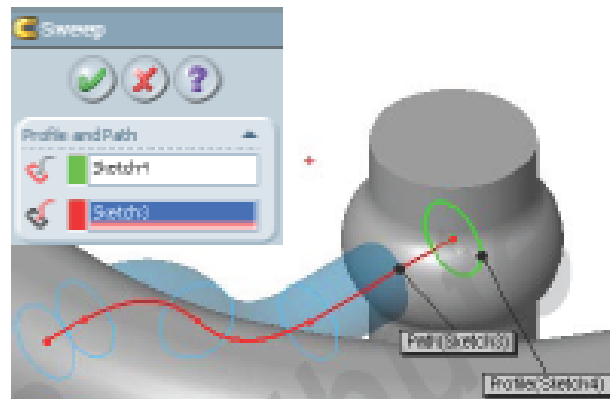
**38 Drag relation.**

Drag the centerpoint of the circle and drop it on the endpoint of the line in the previous sketch. A coincident relation is added between them. Exit the sketch.



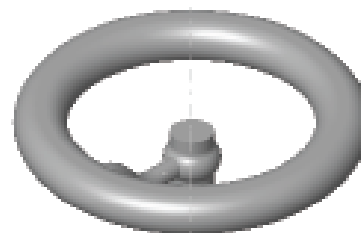
### 39 Sweep.

Click the **Sweep**  icon and select the closed contour sketch as the **Profile** and the open contour sketch as the **Path**.



### 41 Temporary axes.

Display the temporary axes using **View, Temporary Axes**.

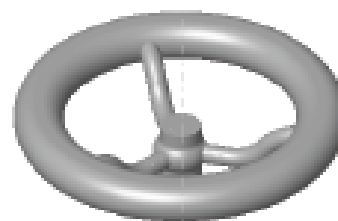


### 42 Pattern the Spoke.

Click **Circular Pattern** . Select the temporary axis as the center of rotation for the pattern.

Click in the **Features to Pattern** list to make it active. Select the Spoke.

Set the **Number of Instances** to 3 with **Equal spacing**.



### 44 Add fillets.

To complete the model, 3mm fillets are added to the highlighted *faces* of the model. Selection of a face selects all edges of that face.

Face selections make the model better suited to withstand dimensional changes.



#### 45 Chamfer.

Add a **Chamfer** feature using the top edge of the Hub feature. Set the distances using the values shown at right.

